Evaluation of barrier effect with ANSAEM

Construction of underground flow barriers can cause increased pressure and consequent mounding upgradient of the structure and drawdown on its opposite side. This lesson demonstrates how to estimate the impact of hydraulic flow barriers, such as sheet piles or cut-off walls that are used in civil construction. Two methods will be presented: the first method is a simplified approach incorporated in Hydrogeologist Workbench, while the second method uses analytic elements module ANSAEM.

1 Case study

A 15 m deep excavation for underground storage facility will cover an area of 50 by 20 m at the location shown on Figure 1. The most permeable top of geological profile consists of 15m of unconsolidated to poorly consolidated alluvial deposits. With the presence of one perennial river (i.e. Main River) and two ephemeral creeks, it is expected that water will seep into the excavation during construction. A barrier is planned to fully penetrate the alluvial aquifer so water seepage during construction can be minimised.

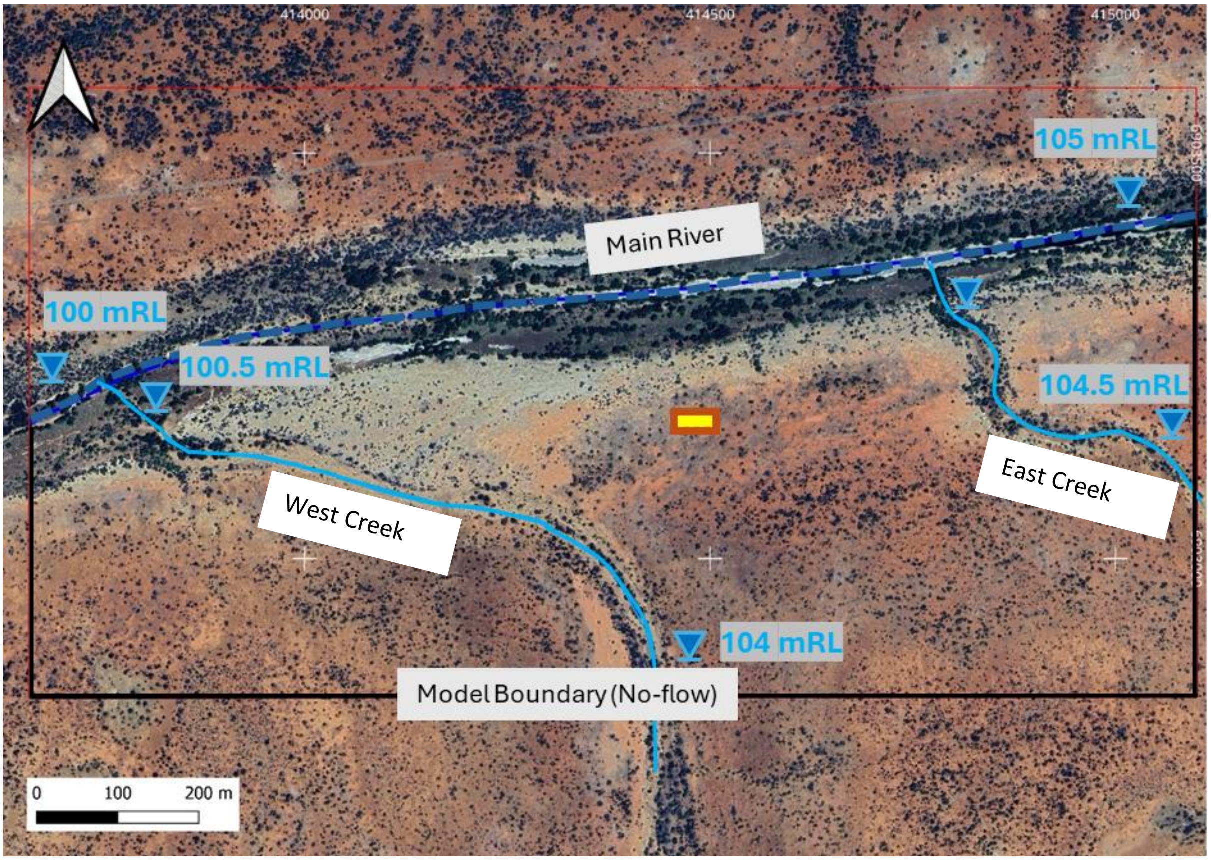


Figure 1: *Location map showing the excavation (yellow) and the barrier (brown, one of the modelled options); the dark blue stripped line indicates Main River (time-constant head boundary); the light blue lines – West Creek and East Creek (transfer boundary condition or "river-boundary in ANSDIMAT); the black lines show the outer model no-flow boundary; the thin red line marks the limit of the project map; numbers on the map refer to water levels in the main river and tributaries.*

This Lesson aims at:

- Estimating the efficiency of the barrier in reducing seepage;
- Assessing impact of different barrier configurations (i.e. full or partial enclosure);
- Assessing impact of barrier on water table mounding.

The following parameters will be used:

- Excavation dimensions: 50 m by 20 m
- Map area: 1440 by 750 m
- Model thickness and barrier depth: 15 m
- Base of the model: 90 mRL
- Top of the model: 105 mRL
- Base of excavation 90 mRL
- Hydraulic conductivity: 1 m/d
- Recharge: 0.0001 m/d
- Main River stages: changing between 100 mRL and 105 mRL (as illustrated on Figure 1)
- Excavation depth: 15 m

2 Simplified calculations using Hydrogeologist Workbench

Go to menu "*Hydrogeologist workbench/Stream/Aquifer interactions*". A pop-up window opens, allowing to calculate pressure changes resulting from structure built across the groundwater flow path.

Select the first tab "*Barrage*" to calculate the barrier effect of a structure.

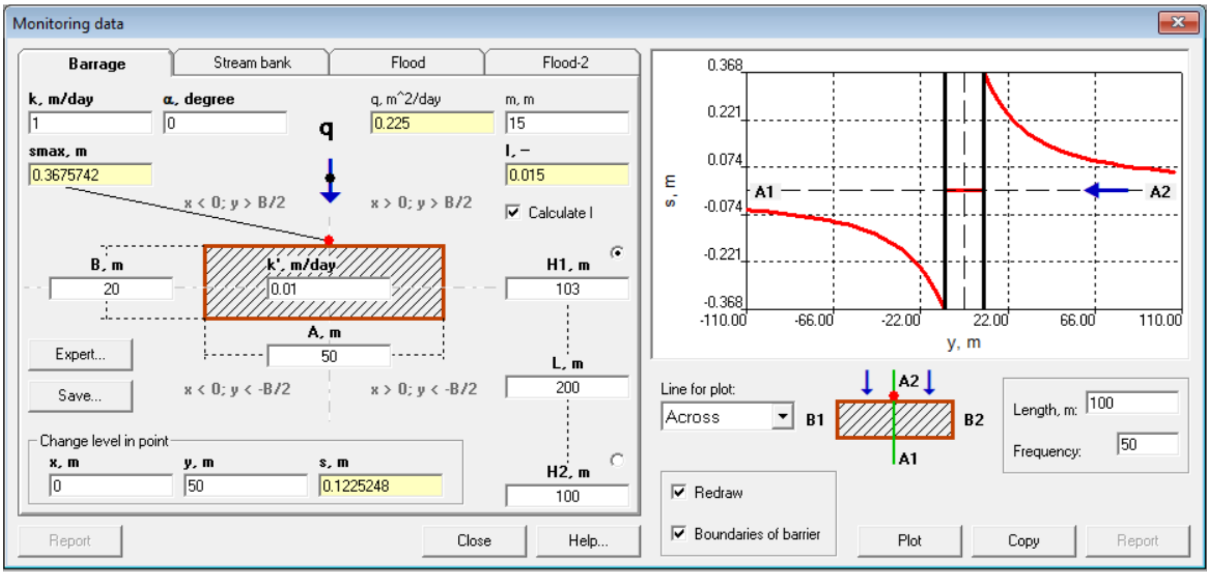


Figure 2: *The barrage tab of the river/aquifer interactions window allows calculating the barrage effect for simple shapes in simple hydraulic configurations*

Enter the following parameters:

- A = 50;
- B = 20;
- k = 1

- Alpha = 0 (angle of hydraulic gradient relative to orientation of structure);
- m = 15 (layer thickness);
- H1 = 103;
- H2 = 100;
- L = 200.

H1 is the distance from the site to the main river and H2 is the distance from the site to the eastern creek.

After entering the parameters, the window should look like *Figure 2*.

The red dot on **Ошибка! Источник ссылки не найден.** indicate the water level at the point of highest mounding. The origin is located at the centre of the diagram. The calculation indicates that a mounding of about 36 cm will occur upgradient, while a symmetrical drawdown should occur down gradient of the barrier. A black point, where drawdown or mounding is calculated, can be placed anywhere on the panel by specifying its coordinates in the fields *x* and *y*.

This solution is assuming a simplified case when a barrier is placed over the entire rectangular area (i.e. excavation filled in with low-permeable material).

For more realistic considerations, a barrier should be modelled as a contour, not as an area/volume. Furthermore, a barrier may be placed on 1 or 2 sides only, not around entire excavation. Next section explains how this can be achieved using the analytic element approach. See ANSAEM: Analytic element models for groundwater flow pattern modelling and interactive help system for information about analytic element methods and how they are implemented in ANSDIMAT.

3 Detailed modelling using analytic elements

3-1 Start a new model

Use the menu *Flow models/Analytic element models/New model* to start a new analytic element model. A new window opens where you can select the folder for the model. Create a new folder, call it "*Lesson 9*", then call the model "*Lesson9*" and save it. It is recommended to save each model in a different folder.

Once a model file is created, the modelling interface opens with a pop-up window where you can input the dimensions of a model. Change default dimensions by entering 1440 for Length and 750 for Width, as in Figure 3. The coordinates of the origin correspond to the lower left corner of the screen, leave them at (0,0) for this example. This window can be accessed later with the "Model" icon  in the Toolbox on the right of the modelling interface.

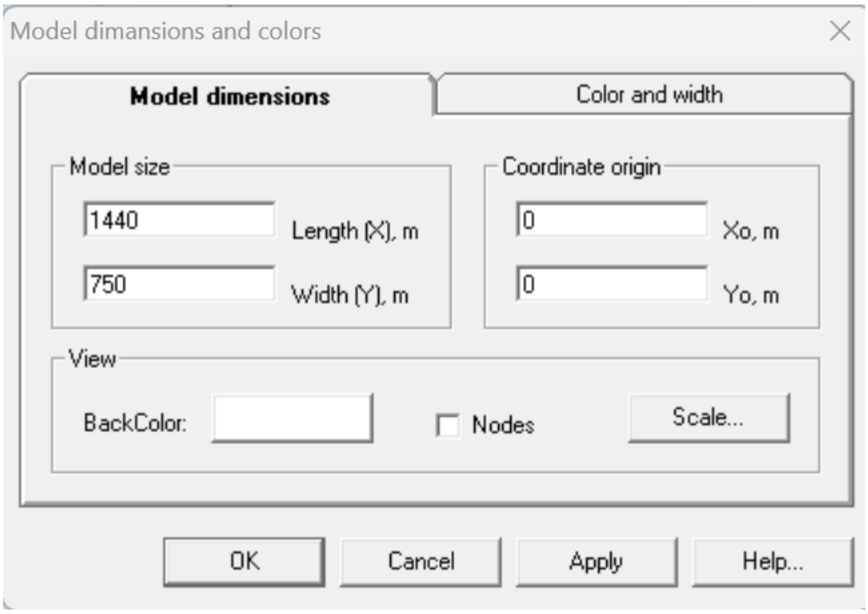


Figure 3: *Model dimension window*

3-2 Aquifer parameters

Use the "*Parameters*" tile in the Toolbox π to assign aquifer parameters. Set the base elevation at 90, the thickness at 15 m and the hydraulic conductivity at 1 m/d (alluvium aquifer). Check the "*Use*" box above "*Recharge*" field, enter a value of 0.001 m/d and define recharge area as the whole model area by entering the coordinates of its lower left corner (x1, y1) and upper right corner (x2, y2). The "*Parameters*" window should look like the Figure 4.

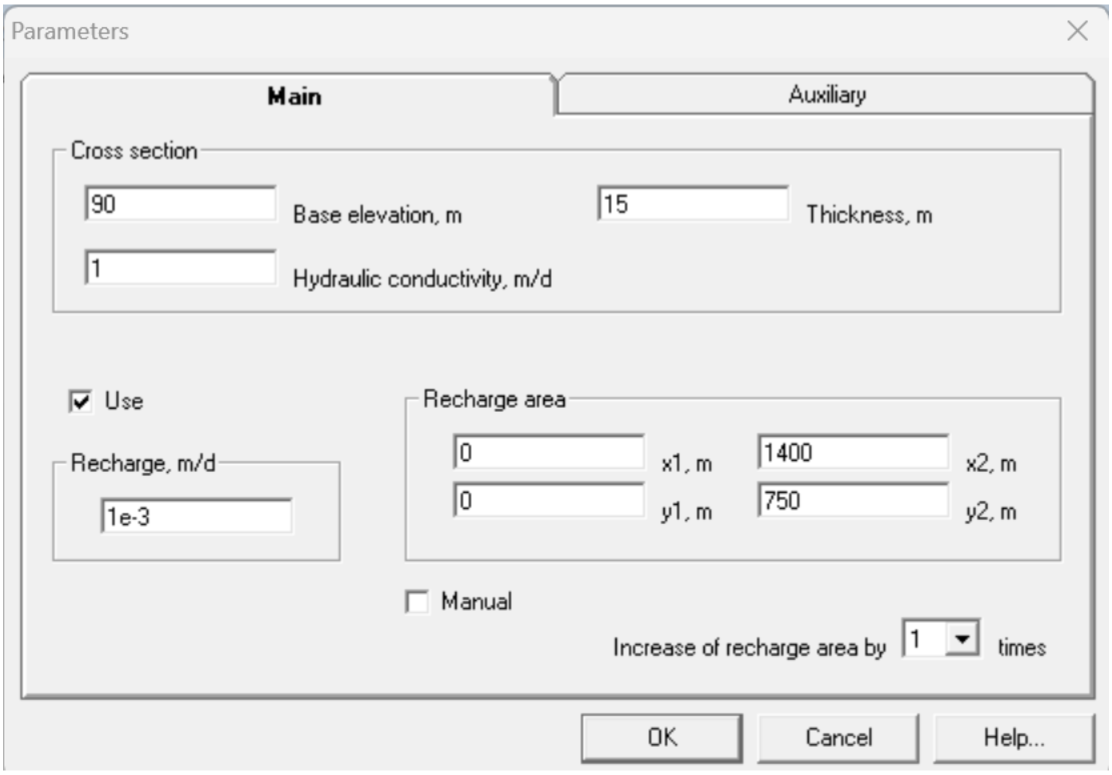


Figure 4: *Parameters window*

3-3 Insert a map

The procedure to import a map is described in details in the Lesson 6. The map for this lesson is included in the Lesson 9 pack. Two files are provided: file “Lesson9_Map_annoted.jpg” includes the location of the boundaries and featured elements, file “Lesson9_Map_clear” only includes the background picture. It is recommended to use the first file to setup your boundaries then change to the second file to run the model and prepare maps for reports.

Click on the “*Load Picture*” tile in the Toolbox, select the corresponding picture and check the box “*Picture visible*”. The window should look like Figure 5. Press the “*OK*” button, and the modelling interface should look like Figure 6.

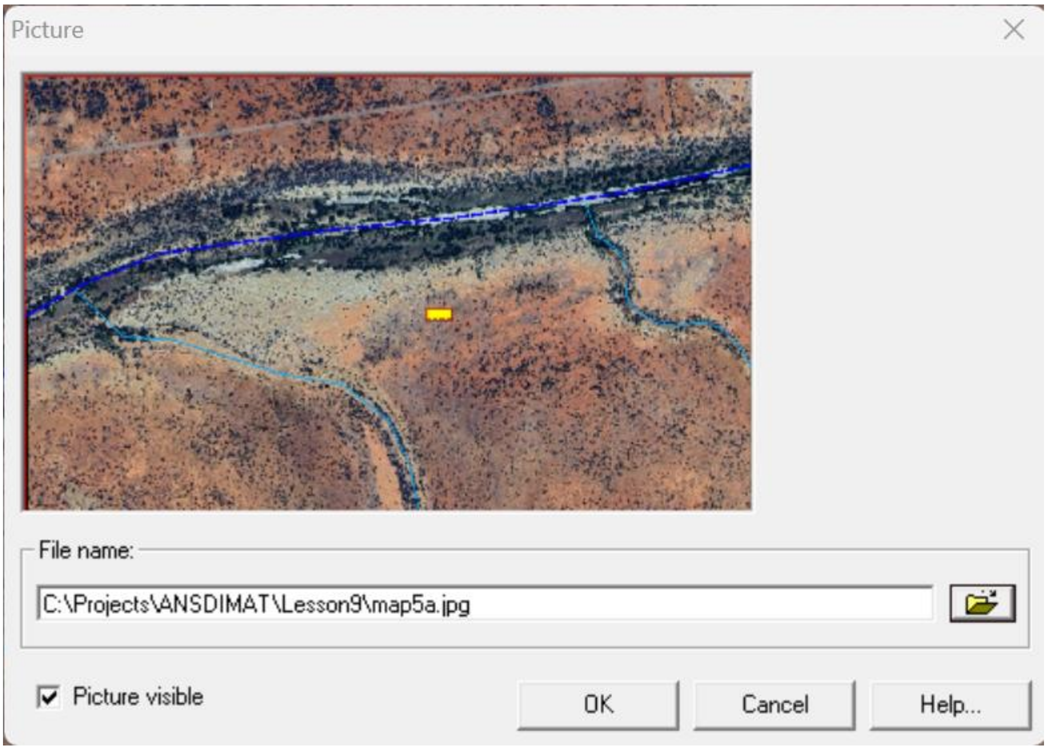


Figure 5: *Picture window with selected image; tick the "Picture visible" to display the map in the model interface.*

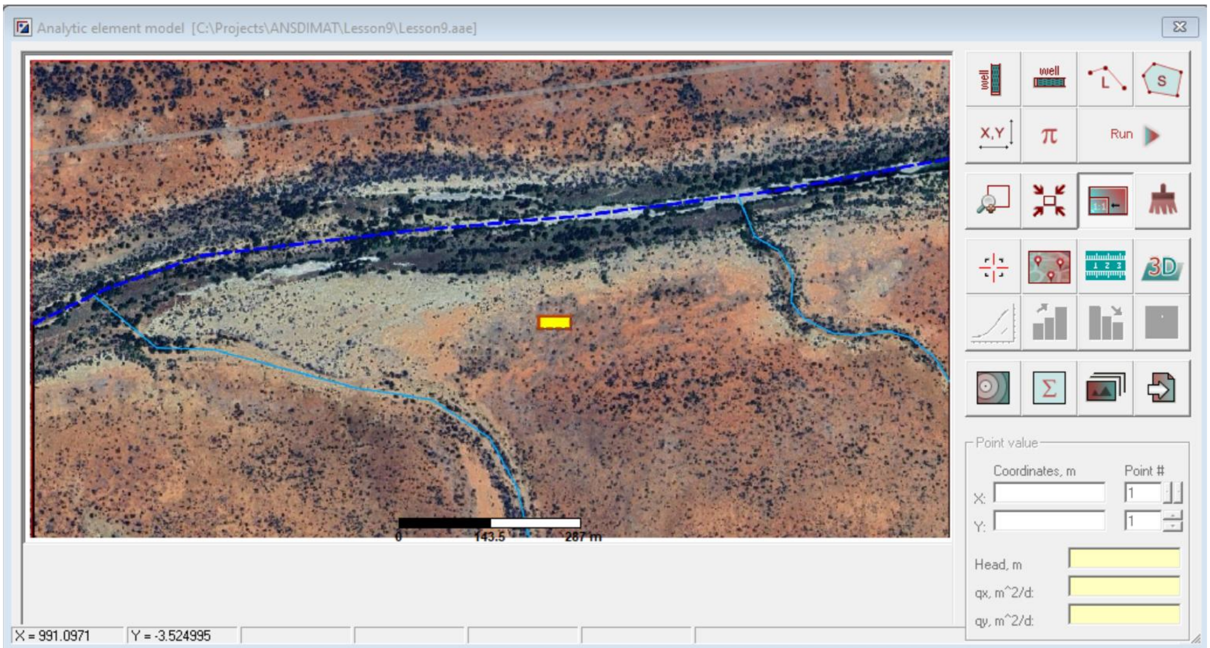
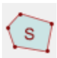


Figure 6*: Model interface with map displayed.*

3-4 Model area

To assign external boundaries, the first step is to outline a contour of a model area. Hydrodynamic boundaries will be later defined along this contour.

Select the "*Polygon*" tool in the Toolbox . In the pop-up window, click on the "*New*" button, then click on the large river (dark blue line) to follow its path across the whole map area, then return to your starting point through the lower corners of the picture (black lines). Do not overlay your last point with your first point, the polygon will close automatically between the start and the end points. Click on the "*Name*" field and change default name to "*Model contour*". Select the "*Model zone*" from the list "*Type*". By default, any created area will be used for the model calculation: an area can be omitted by unchecking the box "*Use area*". The window should look like Figure 7.

As the area is being drawn, a table appears on the right part of the dialog window that shows coordinates of each point. You can adjust the coordinates of a point manually by editing the numbers in the table. These coordinates can be saved by copying the table into an excel spreadsheet using the "Copy" button.

Alternatively, you can paste coordinates from an EXCEL spreadsheet into the table using the "*Paste*" button. Coordinates for all elements of this lesson are provided in the EXCEL file supplied with this lesson.

Once a model area has been defined, the graphic window interface should look like Figure 8.

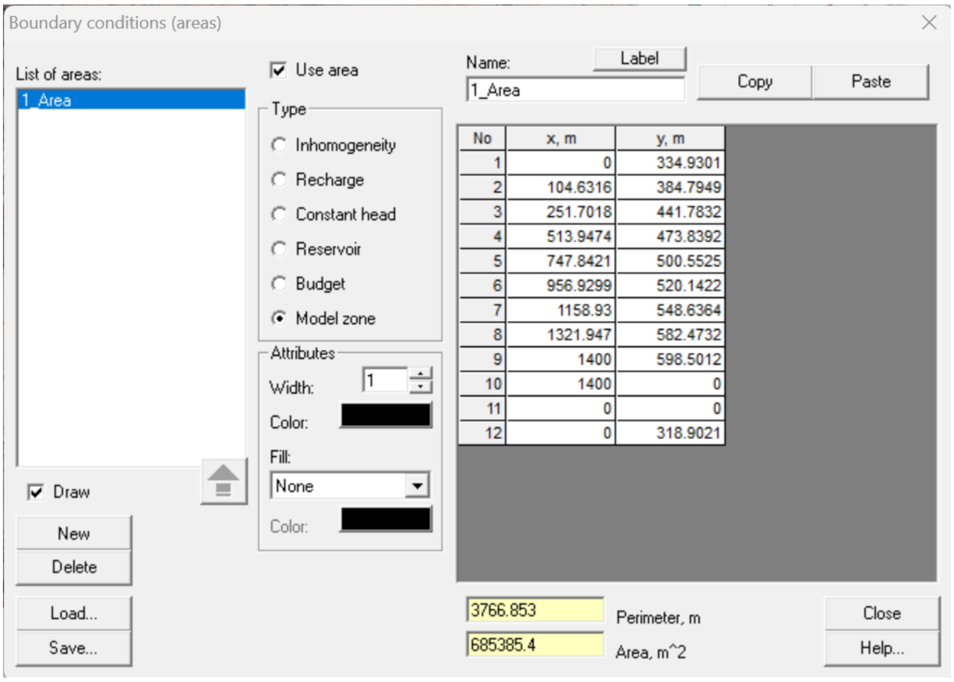


Figure 7: *Boundary conditions (areas) window.*

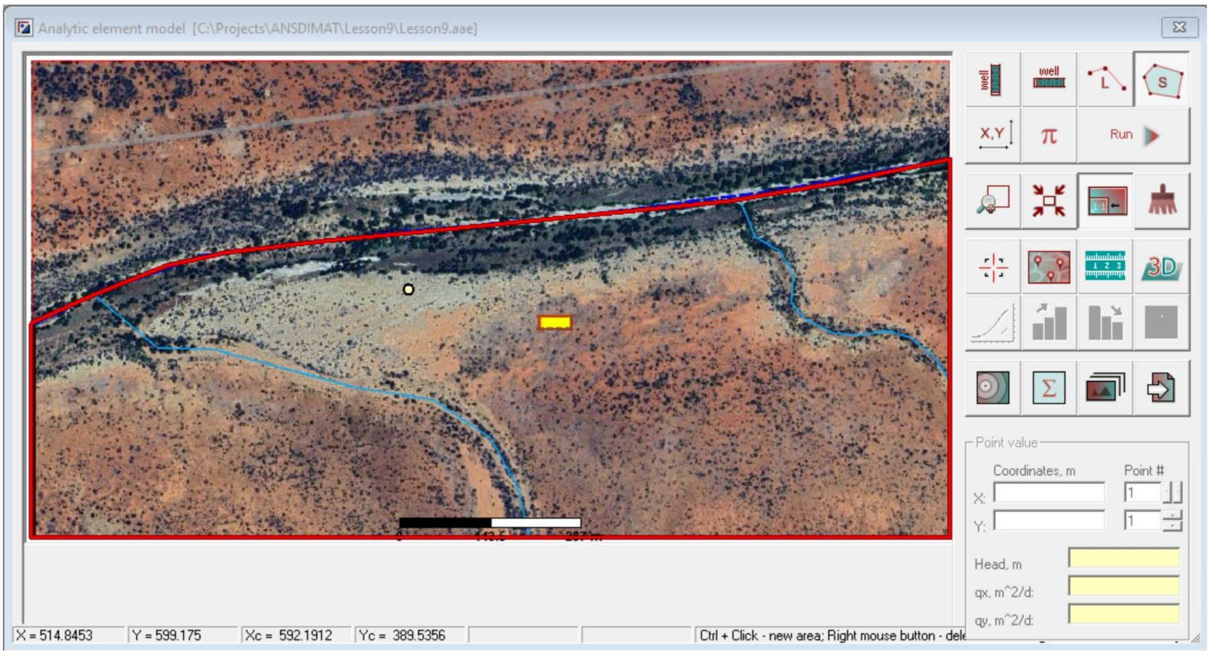


Figure 8: *Model interface displaying the map, the red line marks the newly defined model boundary.*

Note

For efficiency in calculation, define your lines and polygons with the minimum possible number of points. The more points a feature is made of, the higher the longer simulation time is required. High number of points can also negatively affect the model convergence.

3-5 Boundary conditions

Boundary conditions can be generated by two methods: selecting a portion of a polygon and assigning a boundary type and value, or by drawing a new line and defining it as a boundary. For this lesson, we will assign the boundary condition "*Model boundary*" to a newly created polygon.

Open the "Polygon" window by clicking on the icon on the Toolbox. In the table, use the left button of the mouse and press the "Ctrl" button of the keyboard to select points along the main river (Ctrl + Left click). The selected points are highlighted as a green line on the main map. Upon releasing the mouse button, a menu appears, as in Figure 9: this allows assigning a boundary condition to the green line. The main river is defined as the I-st type of boundary condition (Constant Head). Select "*I Kind (H = Const)*" from the list and call it "Main river" in the pop-up window.

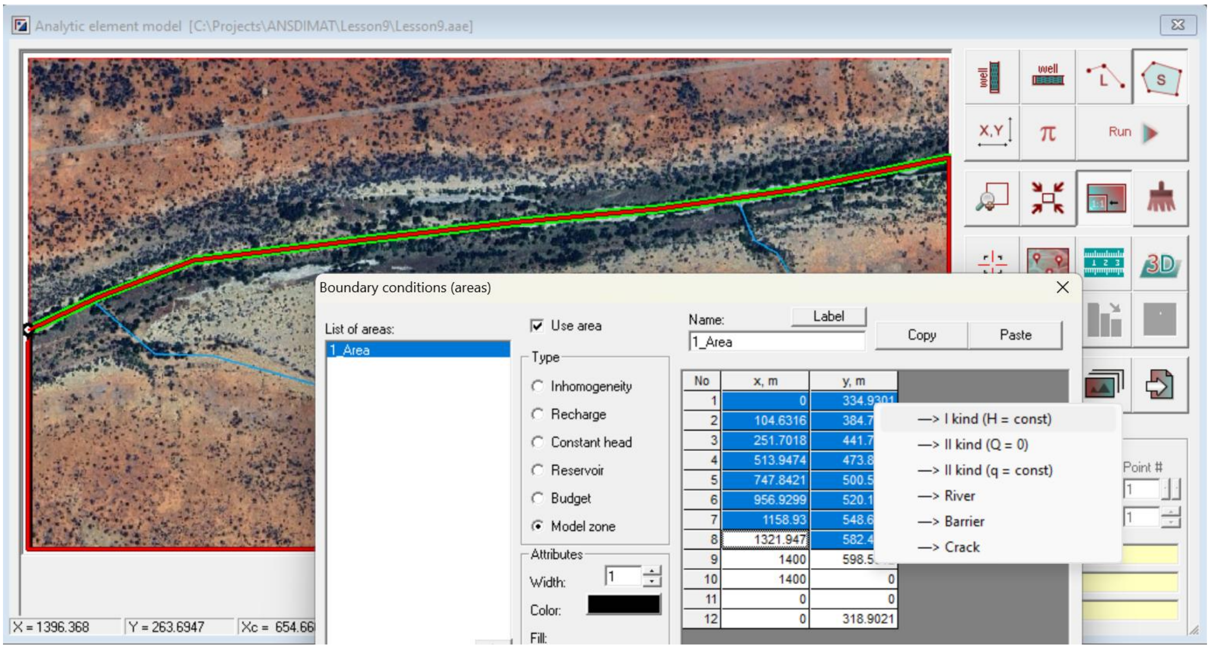


Figure 9: *Boundary condition (polyline) menu with points selected by using "Ctrl + left click" in the table and the corresponding line highlighted in green on the map.*

Repeat the process by selecting the remaining points and assign them to a no-flow boundary: select "*II Kind (Q = 0)*" and call it "No flow". The main map should now look like Figure 10 with a constant head boundary at the north of the model area and a no-flow boundary on the three other sides.

Rivers and creeks can also be modelled in ANSDIMAT as the III-rd type of boundaries (i.e. "*River*" boundaryor transfer boundary). However, an external model boundary is defined along a river, it is better to model it as a Constant Head. When a river crosses the model, it can be modelled as the IIIrd type of boundary.

Figure 10*: Model interface displaying the map with the constant head boundary (blue) and the no-flow boundary (black).*

Click on the "*Line boundaries*" icon in the Toolbox . The "Boundary conditions (polylines) opens, it already includes the two boundaries that were just created. Check that your defined boundaries are called "Main River" and "No flow", otherwise change their names.

The "Main river" boundary has an additional field for the elevation of the constant head. By default, it is set at the base elevation plus the thickness of the model. In this lesson, the river flows from east to west (right to left): **check that the boundary reaches the contour of the model before you interpolate elevation.** X-coordinates of the west and the east ends of Main River should be 0 m and 1400 m respectively, as shown on Figure 11. Correct the coordinates in your table if necessary.

Interpolating elevation is done in the table. Select the elevation in the top row (west end point) and assign to it the value of 100 m, then set the elevation in the bottom row (east end point) at 105 m. Select all the points from top to bottom while pressing the "Shift" button on the keyboard (Shift + left click): the elevation will be interpolated for each point between the two extremes. The table should look like Figure 11.

Let's set up internal boundaries (i.e. creeks) now. Click on the "*New*" button to draw the western creek. Alternatively, you can also copy and paste the coordinates of the points from the excel file, that is supplied with Lesson 9. Once you are finished, select the "River" type. The line colour will turn cyan and two additional columns will appear, one for the elevation and one for the depth of the river. Select the extremes and assign 100.2 m for elevation and 2 m for depth to the left end point. Then assign 104 and 2 for the other end point, for elevation and depth respectively. Finally, select all the rows with Shift + left click to interpolate the elevation and depth between the extremes.

Now that the river profile has been defined, the properties of the river bed must be assigned. In the fields under "Parameters for river bed", assign:

- 0.3 m/day for k (hydraulic conductivity);
- 3 m to thickness;
- 10 m to width.

Once you are finished, the table should look like Figure 12. Enter “West Creek” in the “Name” field.

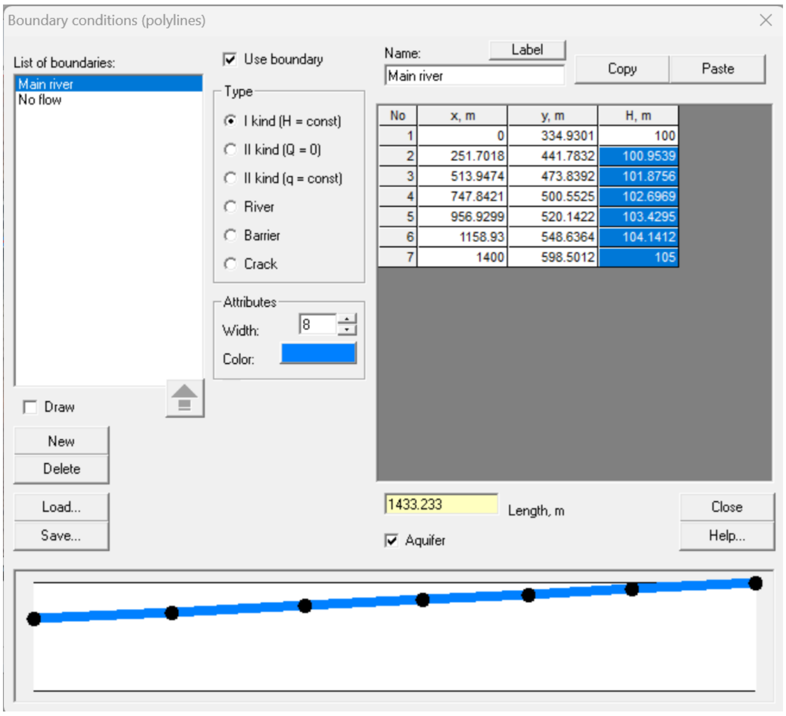


Figure 11*: Boundary conditions (polyline) window with the constant head boundary selected and the associated table.*

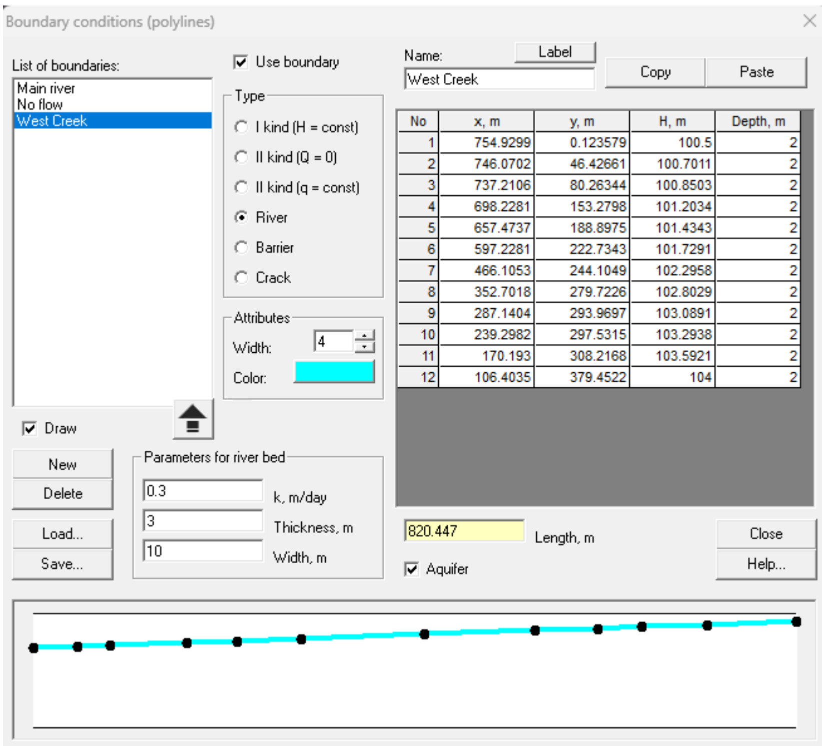


Figure 12*: Boundary conditions for West Creek (polyline) window with the "River" selected.*

Follow then the same procedure to create the eastern creek. Its elevations are 105 m (at the map boundary) and 104 m (at the junction with the Main River) and its depth is 1 m. Use :

- 0.3 m/d for the conductivity of the river bed,
- 1 m for its thickness
- 5 m for the width

Call the element "*East Creek*". The window should look like *Figure 13* and the main map should look like Figure 14.

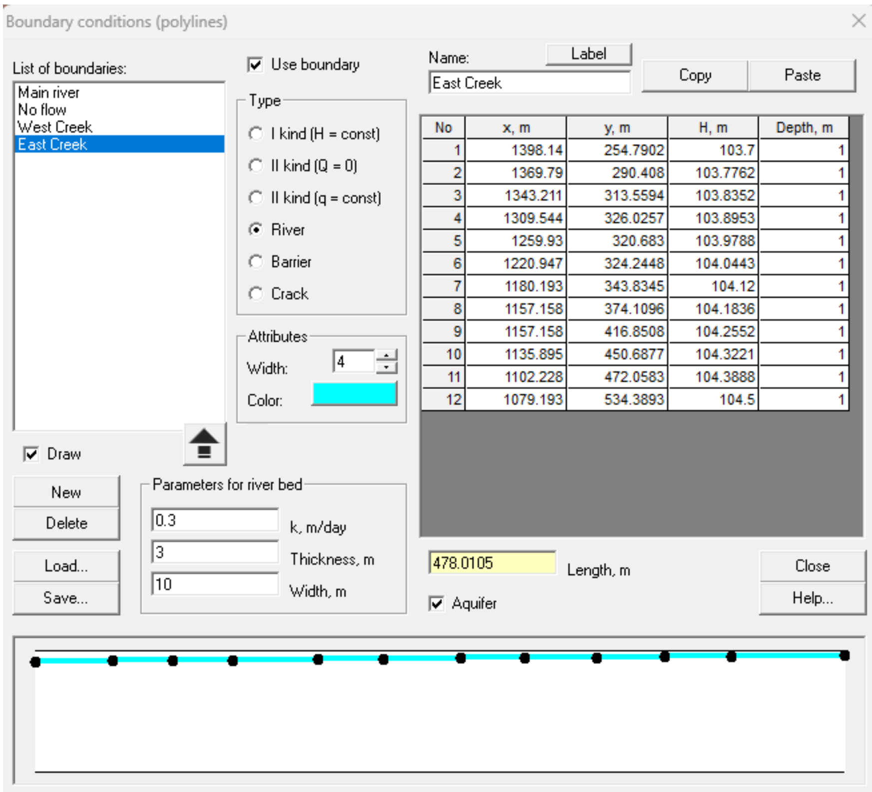


Figure 13: *Boundary conditions for East Creek (polyline) window with the "River" selected.*

Figure 14: *Model interface with the map displaying the "River" boundary condition (cyan).*

At this point, your initial (i.e. pre-excavation) model is setup: if you run it, it will calculate pre-excavation water levels over the model area. For this Lesson, however, we will run pre-excavation model after setting up and deactivating excavation boundary.

3-6 Excavation boundary

An excavation is modelled as a Constant Head area with a value equal to the excavation bottom (in this case, 90 m).

Open the "*Areas*" window by clicking on the icon in the Toolbox. Create a new polygon at the location of the excavation. Set the type as "*Constant head*" and set the elevation H at 90 m.

Then close the areas window. If the feature is too small, use the zoom function  to zoom on it prior to opening the "*Areas*" window. You can also type the coordinates of the points or paste them from this Lesson Excel spreadsheet (*Figure 15*).

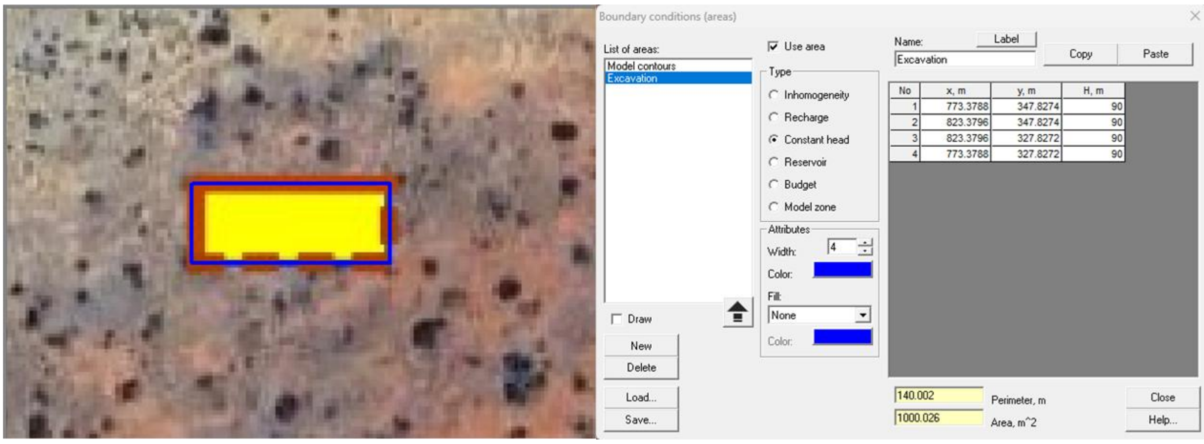


Figure 15*: Model interface displaying the map with the constant head area making the location of the excavation.*

To create a hydraulic barrier (i.e. a pile sheet wall or a cut-off wall), zoom on the excavation with the zoom icon  in the Toolbox and open the *"Boundary conditions (polylines)"* window. Draw the barrier on the northern sides of the excavation then select the type as "Barrier". Leave the parameters as they are (k=0 m/day and thickness = 1 m). The map area should look like Figure 16.

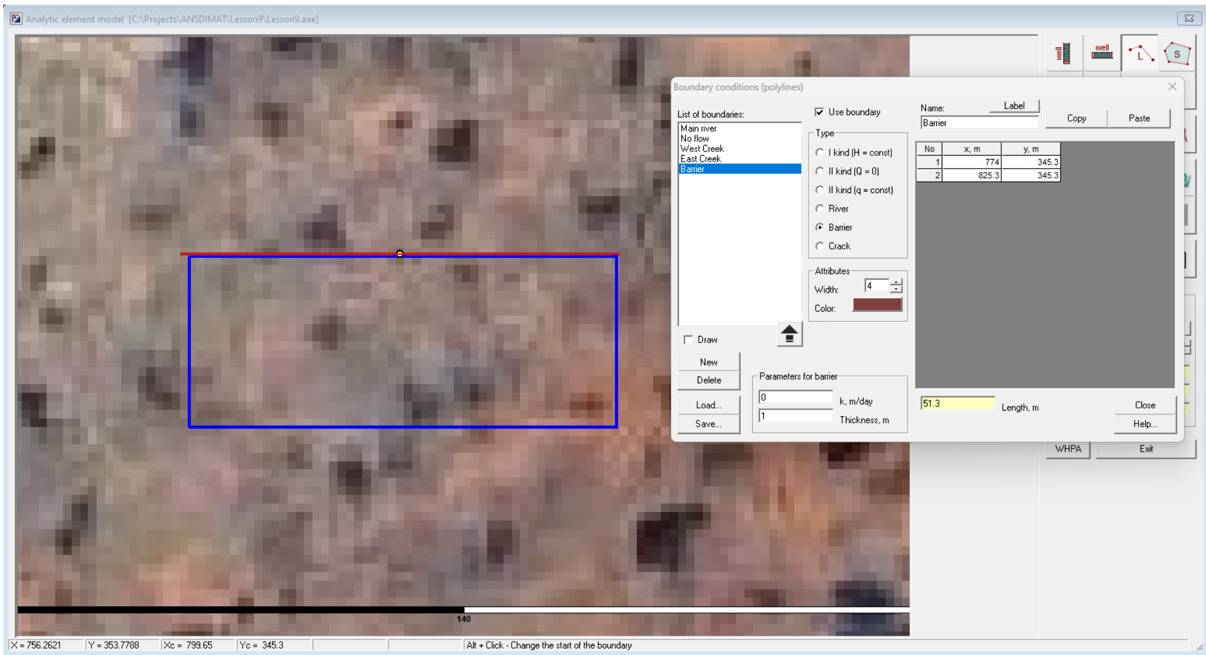


Figure 16: *Barrier drawn on the western and northern side of the excavation*

Notes

In older versions of ANSDIMAT, when two boundaries are overlaid, they may create an issue for simulation. Please make sure you are using the most recent version of ANSDIMAT.

3-7 Running the model

Click on the "*Run*" icon in the Toolbox 

. The "*Run Split*" window that opens should look like the Figure 17.

Since this is a training exercise, change the Potential Tolerance parameter to 1e^{-2} and the Precision to 3. The decrease in precision will increase the computation speed and the decrease in Tolerance will affect the convergence of the model by allowing it to reach a mathematically acceptable solution with a smaller number of runs. You may use this approach to check quickly the validity of your conceptual model but it may be necessary to increase tolerance and precision for the final run.

The "*Run*" button closes the window and starts simulation process. The "*Default*" button allows resetting all parameters to their default values. The "*Close*" button closes the window without launching the simulation. For more information about the parameters in this window, refer to the Help menu accessible through the "*Help*" button.

Once you have set the parameters, press the Run button. A DOS window appears showing the progress of the simulation (Figure 18). The Maxchange parameter should decrease with each iteration if the model is properly converging. A lack of convergence indicates issues with one or several items in the model.

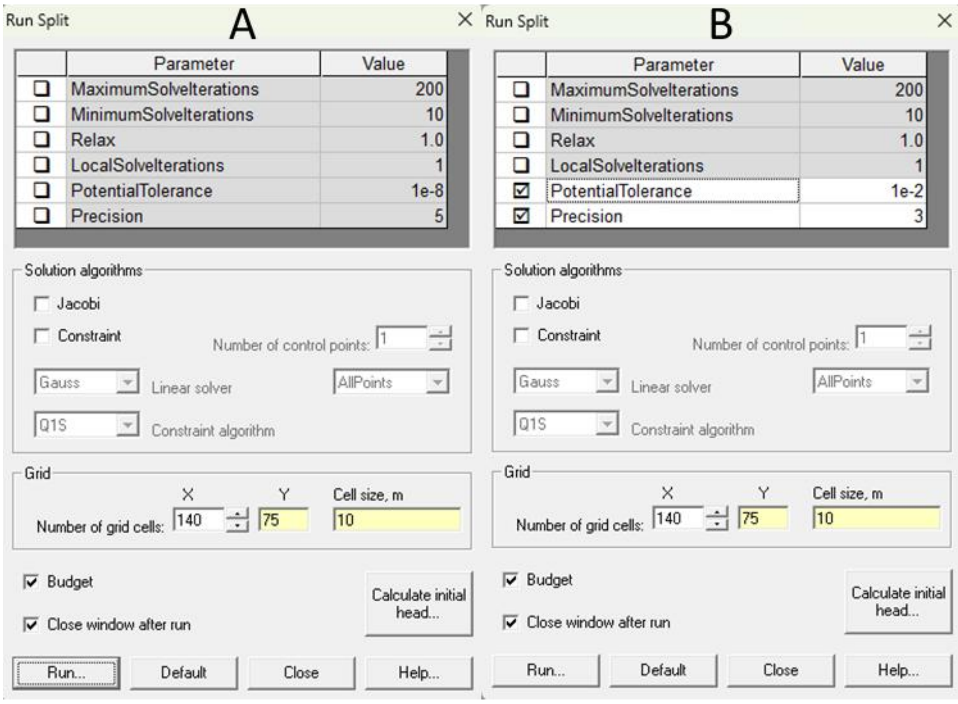


Figure 17: *Run Split window with default parameters (A) and with modified parameters (B).*

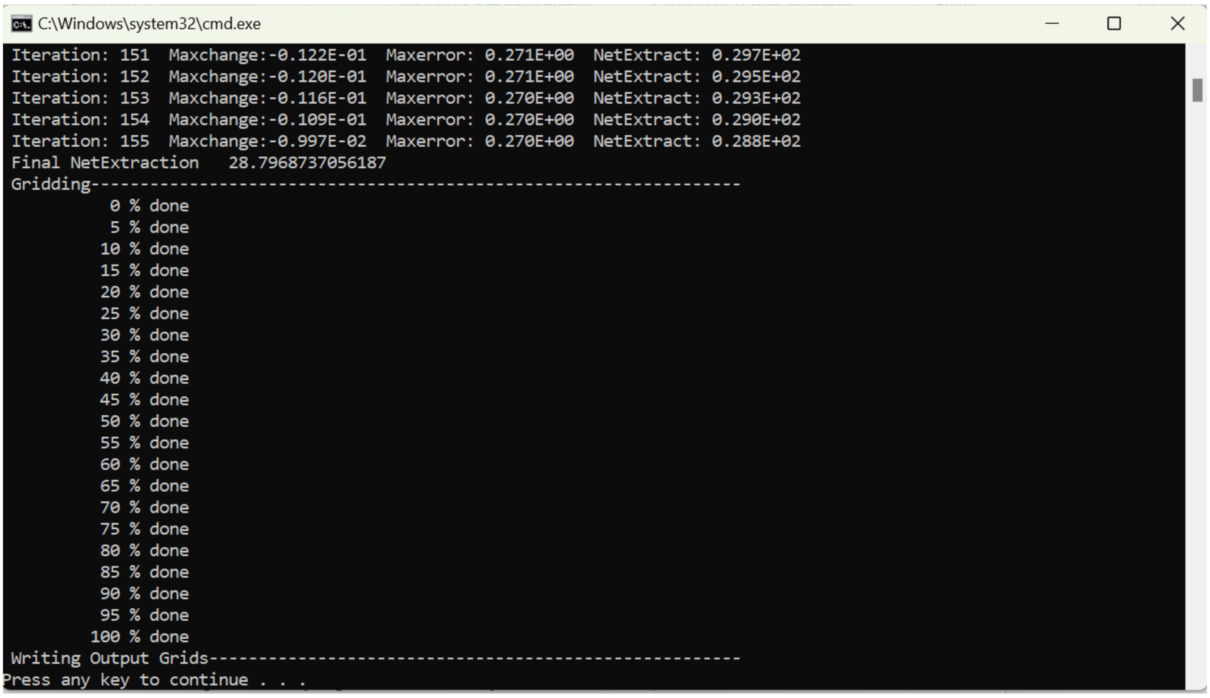


Figure 18: *Split running window at the end of the simulation.*

Note

The model may not converge within the allocated number of iterations. In this case, increasing the number of iterations may allow the model to converge. Relaxing the convergence parameters may also help. If the model still does not converge, you should review your parameters and the shape/complexity of your elements.

3-8 Simulating model in stages

It is recommended to run model in stages, increasing progressively the level of complexity. Once you are satisfied that the pre-construction model results, you can activate the polygon simulating the excavation without hydraulic barrier; after that you can add a hydraulic barrier to a model.

To deactivate the excavation boundary, open the "*Boundary conditions (areas)*" window and deactivate the "*Excavation*" by unchecking the "*Use area*" box, then close the window. To deactivate a hydraulic barrier, open the "*Boundary conditions (polylines)*" and deactivate the "*Barrier*" then close the window.

Click the "*Run*" icon. A console window opens displaying the Split convergence parameters from run to run. Check that the residual difference decreases from run to run. If you changed the Precision and the Tolerance, it should converge in about 155 runs. Once the calculations are finished, press any key to close the window and return to the modelling interface.

Use the "*Potentiometric surfaces, flow velocity*" icon  in the Toolbox to display hydraulic head contours. The "Model results – maps" window has been described in details in Lessons 6 to 8. Click on the "*Values*" tab, click on the "*Find values*" to automatically set the minimum, maximum and step or enter values in the respective fields, then click on "*Fill the table*". The result should look like Figure 19.

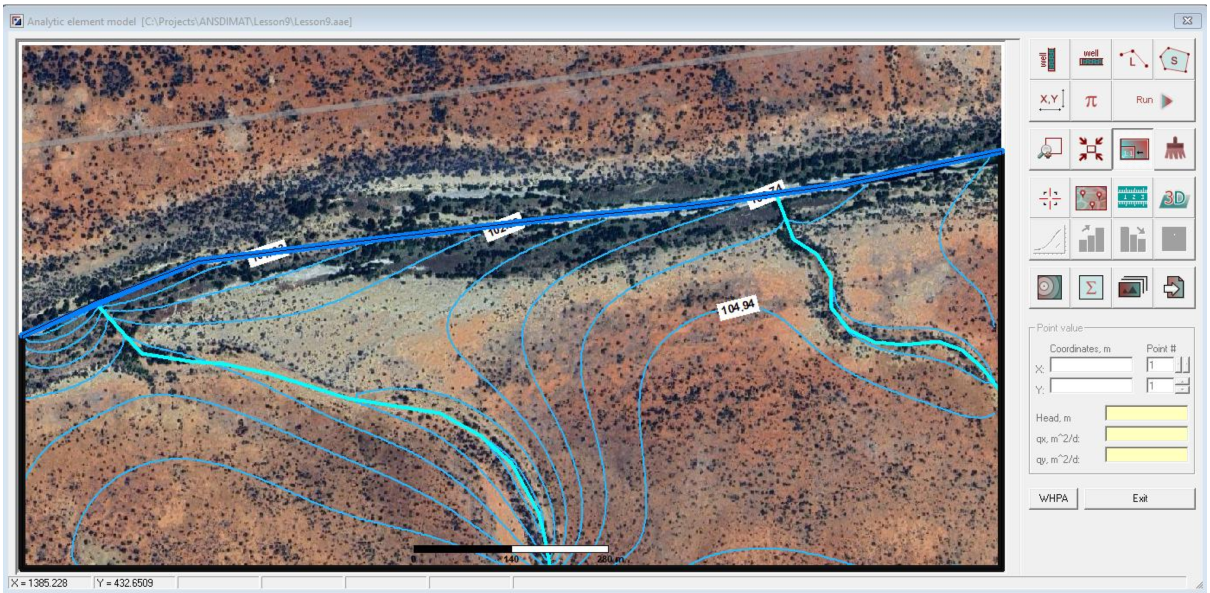


Figure 19: *Contours showing the piezometric heads calculated for the model area without the excavation.*

To activate an excavation boundary, open the "*Boundary conditions (areas)*" window, select "*Excavation*" and check the box "*Use area*". Then run the model again and display the contour lines. The result should look like Figure 20 showing that the excavation is creating a drawdown cone in the aquifer

Figure 20: *Contours showing the piezometric heads calculated for the model area including the excavation.*

Open the "*Boundary conditions (polyline)*" window and activate the "*Barrier*" boundary. Then run the simulation and display the contour lines. The result should look like Figure 21.

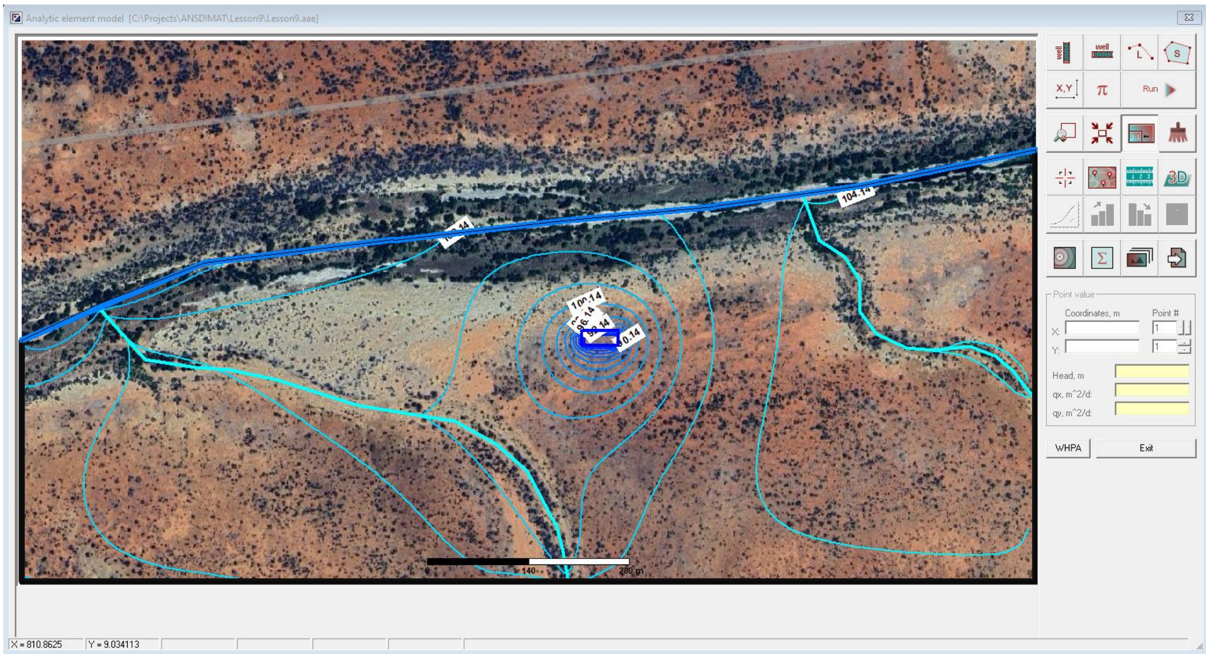


Figure 21: *Contours showing the piezometric heads calculated for the model area including the excavation and the barrier stopping the water inflow from the main river.*

The cone of depression is now flatter as the barrier prevents the water from the river to enter the pit. You can now use the same process to extend the barrier to the western side (to prevent inflows from the second larger river) to see how it affects the shape of the cone of depression.

Note

As complexity of the model has increased, it may not converge within 200 iterations. If this is the case, increase the number of iterations in the “Run Split” window. If the model still does not converge, increase also the precision. In this case, keeping a tolerance of 1e-2, the model meets the acceptance criteria slightly above 200 iterations with a precision of 5. Reducing the tolerance will provide a more accurate result, but will require more iterations as well. You can run at a higher tolerance while improving the model and decrease it for your final run.

3-9. Water budget

To calculate a water budget, the box “*Budget*” must be checked in the “*Run Split*” window (Figure 17). The water budget (Figure 22) can then be displayed by clicking the “*Budget*” icon in the Toolbox Σ. Budget can be calculated for any segment of any object in the model. To evaluate efficiency of the barrier, run the model with the barrier and calculate a budget for the excavation. With a barrier on the northern side of the excavation, the total extraction in equals to about 225 m^3/day (*Figure 22*). With a barrier to the west and to the north, it amounts to about 190 m^3/day, and to about 156 m^3/day with barriers to the west, north and east.

Now deactivate the barrier in the “*Boundary conditions (polylines)*” window and run again the model: the total extraction for the “Excavation” is now 250.1 m^3/day. You can see, that installation of the barrier on two sides of the excavation decreased groundwater inflows by about 20%.

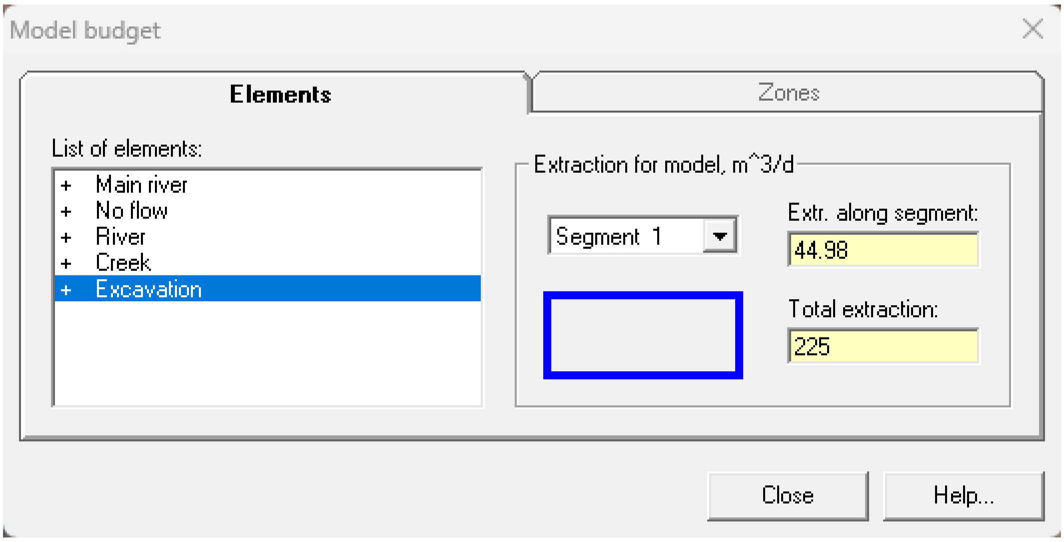


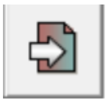
Figure 22: *Budget window, the "Budget" box in the "run Split" window must be ticked for the budget to be available; select a boundary in the list on the left to see the amount of extraction associated with the selection.*

3-10. Displaying drawdown

The ANSAEM module has capability of post-processing model results. For example, you can add or subtract two piezometric surfaces and display the result on a map. If you select the piezometric surface calculated without the excavation as a reference and subtract the surface calculated with the active excavation, the result will excavation-induced drawdown of groundwater table. Similarly, by plotting a difference between water table calculated with and without the barrier, you can illustrate impact of the barrier.

To calculate a drawdown, two inputs are required: one corresponding to the initial conditions, pre-disturbance, and the second - to the post-disturbance hydraulic heads. There are two ways in which drawdowns can be calculated:

- Method 1. Use the model to simulate a pre-disturbance piezometric surface. Save it under a specific name and then recall it in postprocessing, when simulating the post-construction stage.
- Method 2. Use the "*Calculate initial heads*" to store the initial piezometric surface into a temporary file that can be used in postprocessing.

In Method 1, we use the "*Run Split*" button to open the run interface and process the simulation. Then display the result on the map using "*Piezometric surface, flow velocities*" window. A simulation result can be saved by using the "*Save values / picture*" icon  in the main toolbox. In the pop-up window, select a location (the model folder is recommended) and enter a name to the file, for instance "*Initial*". When simulating the post-construction scenario, the "*Initial*" file can be recalled and used in postprocessing. This process is described later in this section.

In this lesson, we are using Method 2: click on the "*Run Split*" button to open the run interface (Figure 17) and click on the "*Calculate initial heads*" button in the lower right corner of the window. A new window appears, where elements for calculation of the initial piezometric surface should be selected. Selection of elements is done by checking boxes against each selected element: by default, all elements are selected. Uncheck boxes against the barrier and the Excavation, as in Figure 23, then click on "*Run*". Split will calculate the surface corresponding to the selected parameters and store it in a temporary file.

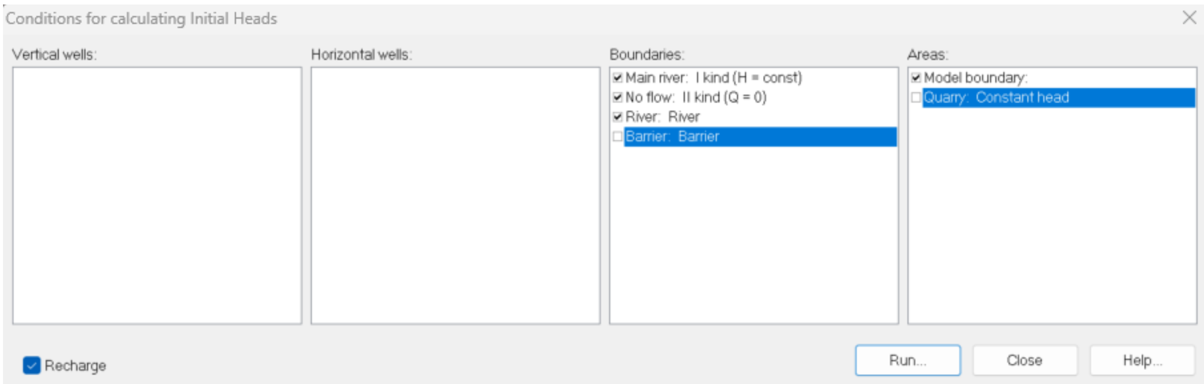


Figure 23: *Parameter selection to calculate initial heads*

Once the initial heads are calculated, the original "*Run Split*" windows reappears: click on "Run" to calculate the water levels corresponding to the post-construction phase.

Use "*Piezometric surface, flow velocities*" window for results postprocessing. Select the "*Values*" tab and click on the "*load Matrix*" button. A new window appears offering several choices (Figure 24):

- you can load a saved matrix (if you have calculated your reference surface and saved it separately, use this option),
- change the matrix being displayed or
- load a predefined operation (drawdown of initial heads); check this option if you used the "Calculate initial heads" process.

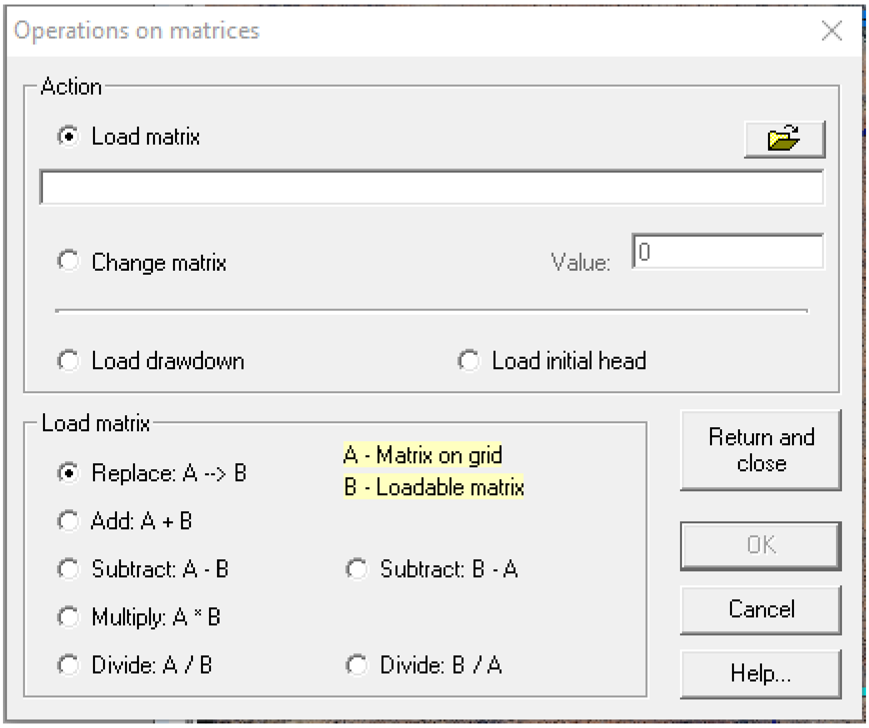


Figure 24: *Option for calculations on matrices*

Use the first option if you have calculated your reference surface and saved it separately. Then, select the operation to apply between the matrix that was just calculated and the matrix that was just loaded.

If you have calculated the initial heads at the same time as the current model, select the option "*Load drawdown*": the current model will be automatically subtracted from the initial heads. The contours will represent the difference between the two matrices: positive numbers indicate drawdown, negative numbers indicate mounding compared to the reference piezometric surface.

Figure 25 shows some mounding next to the location of the barrier. This indicates that the barrier is effective against infiltration from the main river, which is consistent with the comparison between the budget for the excavation and the budget for the excavation and the barrier showing a 20% reduction in inflows with a barrier facing the main river.

Figure 25: *Contours showing the drawdown calculated with initial heads set without the excavation and the current model including the excavation and the barrier.*